# Richard Nakka's *Experimental Rocketry* Web Site

## Machining of Rocket Nozzles

- Introduction
- Materials for Rocket Nozzles
- Use of Blueprints
- Tools required
- Latheing Processes
- General LatheingTips
- Machining the Nozzle
- Machining Graphite
- Safety
- Appendix A -- Custom made Boring Tools
- Appendix B -- Reaming
- Appendix C -- Useful Links
- Appendix D -- Example Nozzles
- Appendix E -- Tapping misadventures
- Appendix F -- Snap Ring Grooves NEW


## Introduction

I learned the basics of using a metal lathe while attending high school, in the "metal shops" course, which was part of the "industrial arts" curriculum. In fact, I made one of my first rocket nozzles using a school lathe, with guidance from my shops teacher who taught me how to carry out the more difficult aspects of latheing, such as boring out the interior cone. Some years later, my brother purchased a small metal lathe for our home workshop, a compact Austrian made *Edelstaal* model with a 10 inch swing, and 24 inches between centres. Using this machine, I progressively improved my machining skills and successfully produced a number of nozzles such as those for the *A-100* , *B-200* and *C-400* motors. I've used this same lathe, in more recent times, to produce the *Kappa* nozzle, which was later used for the *Cirrus One* rocket. A few years back I purchased a metal lathe for my own workshop, a medium sized Chinese made *Craftex* model, with a 12 inch swing, also 24 inch between centres. Being larger, and having a more powerful motor, as well as power cross-feed, makes for more expedient work of producing a nozzle. The largest nozzle I've made to date is that for the 75 mm *Lambda* motor. All the nozzles that I have produced have been made on a manual lathe, with one exception. While attending University, I had the opportunity to program and run a small CNC lathe to make an experimental "cubic" nozzle featuring a well-rounded internal profile. Made for the *B-200* motor, it was static fired to compare the performance with a standard "conical" profiled nozzle. This study formed part of my graduation thesis.

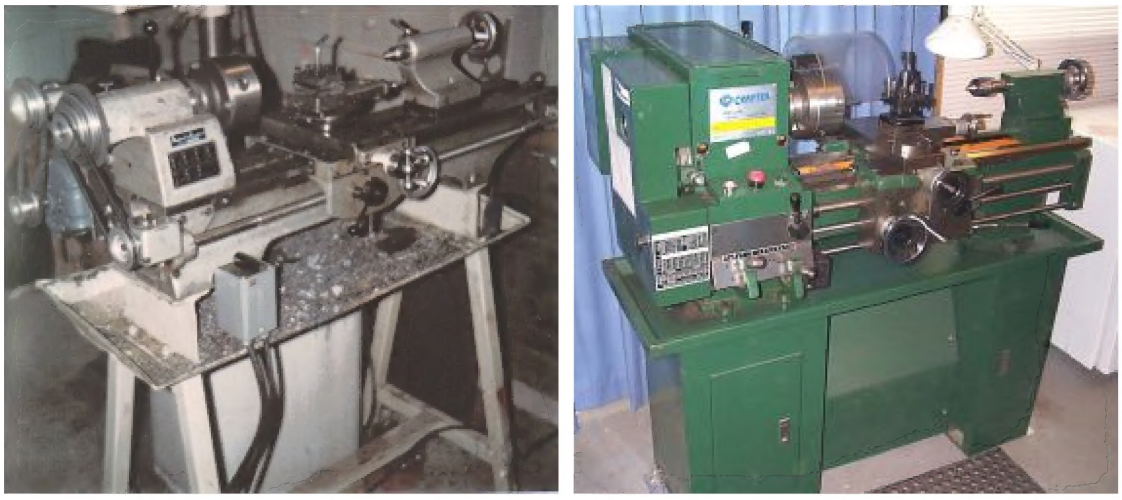


Figure 1 -- ***Left***: The *Edelstaal* lathe ***Right***: My *Craftex* lathe

Without a doubt, machining a rocket nozzle is a challenge, especially for those who have had limited experience in metal working. Machining a nozzle draws upon some of the more difficult aspects of lathe use, owing to the rather complex shape and need for accuracy, combined with the desire of producing a lightweight nozzle having all or most excess material cut away. Machining a nozzle is time consuming, especially if using a small lathe to make a large nozzle. For example, the *Kappa* nozzle took me 18 hours of machining time. More recently, with improved techniques, greater experience under my belt and a larger lathe, an average size nozzle can be completed in 4 or 5 hours.

Still, I'm very much an "amateur machinist". Much of what I have learned regarding latheing techniques have been gained from hands-on experience, from experimenting with different machining techniques, from books, and, unavoidably, from making mistakes. The process of nozzle machining presented here is a compilation (or perhaps distillation) of this experience, using methods that work reasonably well for me. There are undoubtedly variations that can be used successfully and as such, what is presented should be taken more as a guideline than as a set methodology. The means of "best" producing a nozzle is sure to be one of continuing evolution for each individual, including myself. One should feel free to experiment and any suggestions for improvements or simplifications are always welcome.

## Materials for Rocket Nozzles

The nozzle represents a key part of a rocket motor. This component serves to accelerate a voluminous mass of stationary combustion gases to supersonic velocities within a very short distance, and in doing so, produces useful thrust. In accomplishing this remarkable feat, a nozzle is subjected to very high pressures, and rapid, dense gas flow at high temperatures. As such, a nozzle must be fabricated from a material that will be capable of withstanding such conditions of structural and thermal loading. As well, any restrictions to the free flow of gases must be minimized, which necessitates a smooth, suitably profiled flow surface. Considering the effort and time invested in making such a nozzle, multiple usage is obviously desired. Fortunately, ordinary (mild) steel is well suited to this application, at least, for many popular amateur propellants. Mild steel machines relatively well. Either "hot rolled" or "cold rolled" steel may be used. Hot rolled is softer and as such cuts more readily. Cold rolled tends to produce a better surface finish with less effort and produces less problematic chips, or *swarf*. I usually use AISI 1018 steel, perhaps the most readily available grade.
One possible alternative is "Ledloy" (12L14) which is a steel alloyed with a small percentage of lead. I've recently tried machining a nozzle with this steel, and it cuts like a dream. Perhaps the nicest aspect of machining ledloy is that the swarf is in the form of chips which fall away from the cutting tool, as opposed to the long "thready" swarf that mild steel (such as 1018) forms. I have not, to date, fired a motor with this nozzle, so the issue of heat resistance is still an open one. Ledloy is

readily available, but may cost slightly more than regular steel. Another alternative is stainless steel. Both alternatives should be used with caution as these alloys may be less resistant to high temperatures than mild steel. Alloy steel such as 4130 should not be used, due to its lower melting point and difficulty in machining.

---

Update: *May 29/05*
The nozzle for the *Liberty* "L-class" rocket motor was recently machined from 12L14 steel. The intent was to reduce the labour required to machine this rather large nozzle, and this was very much the case. After three firings, the nozzle showed zero throat erosion, so the capability of 12L14 steel to stand up to the heat of combustion has been confirmed.

---

An alternate *non-metal* material that is especially suitable is graphite, although the use of this material requires special considerations, due to the copious amount of messy and electrically conductive dust that is generated. An old shop-vac with a very fine dust filter works well to draw away the dust as it is produced. Graphite nozzles are ideal for use with high combustion temperature propellants such as APCP, for which steel is not suitable.
Low melting point alloys such as aluminum and brass are not suitable for multi-usage nozzles, unless a high temperature resistant "throat inset" is used (which complicates the manufacturing). This technique has been used with good success for Hendrik Lau's *HL6000M* rocket motor. The nozzle is fabricated from aluminum alloy and features a high-density graphite insert (see *Appendix D* for photos).

## Use of Blueprints

It is, of course, quite feasible to make a nozzle "on the fly" without need of an engineering drawing, commonly referred to as a "blueprint". However, the more rational approach to rocket motor design and construction involves designing a nozzle to achieve a particular performance goal. This design process defines the nozzle shape and dimensions. Key design parameters for a standard conical deLaval nozzle are the convergent and divergent angles, and the diameters of the inlet, throat, and exit. As well, the exterior profile of the nozzle is sculpted to minimize the mass of the finished nozzle. A nozzle that does not have a sculpted exterior will work nearly as well. However, if it is made of steel, it will be heavy! Graphite nozzles do not require much or any sculpting of the outer profile, as graphite is very lightweight, having approximately half the density of aluminum. Other design details include determining the diameter of that portion of the nozzle that interfaces with the casing, and o-ring groove dimensions and location. Additionally, the means of attachment to the motor casing, which will typically be machine screws or snap rings. It is important to recognize that the more care and effort that is put into creating an accurate and detailed blueprint, the less laborious the effort to fabricate the nozzle and the less likelihood of making a mistake during machining which could relegate hours of effort to the scrap bin. CAD software programs (even basic ones) are ideal for producing a suitable nozzle drawing. I use QuickCAD, which I find to be easy to use, powerful and relatively inexpensive. Freeware,"Shareware" or free-trial CAD programs are also available. Alternatively, scaled drawings may be made the old-fashioned way, by hand.

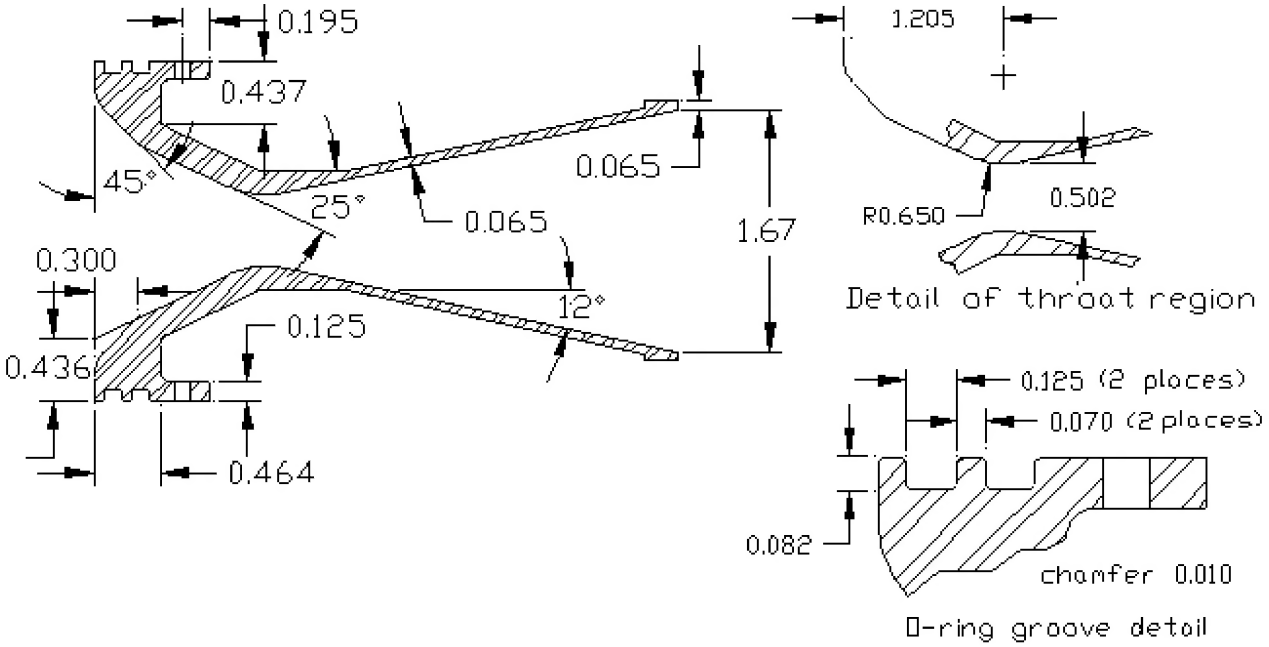



Figure 2 -- Blueprint of the *Kappa* nozzle

## Tools required

In addition to a metal lathe and basic cutting bits such as those required for the turning and facing operations, a few other tools are needed for nozzle making. Boring bits are needed for cutting the interior convergent and divergent profiles of the nozzle. For machining nozzles with a small throat diameter, such as the *A-100*, it will likely be necessary to custom-make a boring tool, as commercial boring tools are generally too large to permit boring of such small hole diameters. A parting tool bit and holder is needed for cutting o-ring grooves and snap-ring grooves. A tailstock mounted drill chuck is also required, as well as drill bits. These are needed for drilling out the throat and for initial material removal of the convergent and diverent cones. A special short, stubby drill bit called a *centre bit* is required to ensure that the drilled hole is perfectly concentric (regular twist drill bits are too flexible). A hacksaw, bandsaw or friction blade is needed to cut the steel bar to length. One other tool is essential - a good quality dial caliper (or digital caliper) gauge, used for precision measuring. Other tools that are useful but not essential are a depth gauge and bore gauges.
Three custom made boring bits suitable for machining nozzles with small diameter throats are described in Appendix A.

## Latheing Processes

Machining a nozzle involves six cutting operations inherent to latheing: **facing**, **tapering**, **turning**, **boring**, **drilling** and **parting**.
The first four of these, as well as other latheing operations, are shown below in Figure 3.

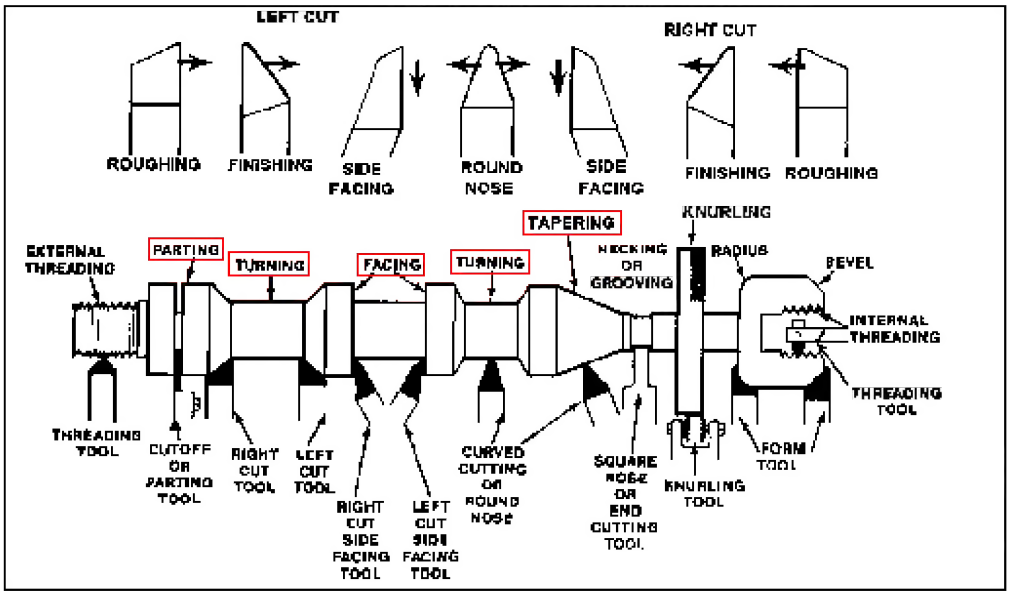



Figure 3 -- The various machining operations applicable to a lathe.

**Facing** is used for truing the ends of the workpiece and to trim the workpiece to the required length. **Tapering** is the operation that forms the outer profile of the nozzle divergent cone, and is performed by rotating the compound-slide at the angle required to produce the divergent cones. **Turning** is the operation that cuts parallel to the workpiece axis and serves to reduce the diameter of the workpiece. **Boring** is an operation that shapes the interior profile of the nozzle, and is the trickiest and most time consuming process. However, much of the boring can be eliminated by first "step drilling" the workpiece to the approximate profile of the interior, then employing the boring operation for the final finishing cuts. The **parting** operation is used solely for creating the o-ring or snap-ring grooves, and as such, is not a true parting process. However, the basic concept is the same.

Another machining process that I've recently experimented with in regard ot nozzle fabrication is **reaming**. A reamer is a tapered cutting bit that can be used to machine the profile of the convergent or divergent cones. The use of a reamer is discussed in Appendix B.

It is highly suggested to review the links provided in **Appendix B** to become more acquainted with the construction and terminology of a metal lathe, as well as the various machining operations.

## General Latheing Tips

Using the correct cutting speed is important in order to avoid "chatter" and to avoid overheating the tool bit (with resultant premature dulling). A higher cutting speed can be used with carbide bits. For basic cutting operations such as turning or facing, a good surface cutting speed for AISI 1018 steel is 130 fpm (ft. per minute) for HSS bits, 1.5 to 2 times that for carbide bits. Ledloy 12L14 steel reportedly can be cut at 325 fpm with HSS bits. Table 1 correlates surface cutting speed to workpiece diameter and RPM.

---

**Example**: A nozzle is being machined from 1 inch diameter cold rolled 1018 steel using a HSS cutting bit. As such, a cutting speed of 130 fpm will be used. From Table 1, the RPM should then be set for 479 RPM (or thereabouts).

---

Feed rate of the cutting bit , and depth of cut, are also important. Feed rate is the linear speed at which the cutting tool is fed along the workpiece; depth of cut is how much material is removed with each pass of the tool bit. Basic cutting operations such as turning can tolerate a greater feed rate or cutting depth than the more delicate operations such as boring. For turning, the ideal cutting rate should produce swarf that gets hot enough to produce a straw-yellow oxidation coating. Blue coloured swarf indicates too high a cutting rate. Lighter cutting rates are fine, but extends the overall time required to fabricate a nozzle. Here is a photo of some first-rate swarf from a boring operation.

Cutting oil (coolant) should be used with HSS (High Speed Steel) tool bits. I use water-soluble coolant, mainly because it cleans up more readily, is odourless and is smokeless. Use plenty of cutting oil when drilling and tapping. Tungsten carbide bits should be used without coolant. Both HSS and carbide bits should be sharpened regularly. This will noticeably ease the cutting operation as well as provide for a better surface finish. Carbide dulls at a much slower rate, however, the cutting edge will gradually deteriorate. For HSS bits, use a regular aluminum oxide wheel for sharpening. For tungsten carbide, a green silicone carbide grinding wheel works well (do not use for HSS bits).

## Machining the Nozzle

The basic steps that I follow for the machining of a nozzle are given below.

1. **Measure and cut** the round steel bar stock to the required length, providing a slight amount of extra length (say, 1/16" , 1.5 mm) in case the cut is not completely square.

2. The workpiece is then chucked in the lathe, and **both ends are faced**, or "trued" such that the ends are completely straight and squared. Enough material should be removed during facing such that the workpiece is of a length equal to the final nozzle length.

3. The workpiece is next **"trued" on the outer surface**. Chuck the workpiece such that slightly more than ½ the length is protruding. Turn off enough material from the outer surface such that the workpiece rotates completely true (remove, say, 10 or 20 thou*). Remove the workpiece and chuck it the other way, and repeat the truing operation. The truing operations are illustrated in Figure 4.

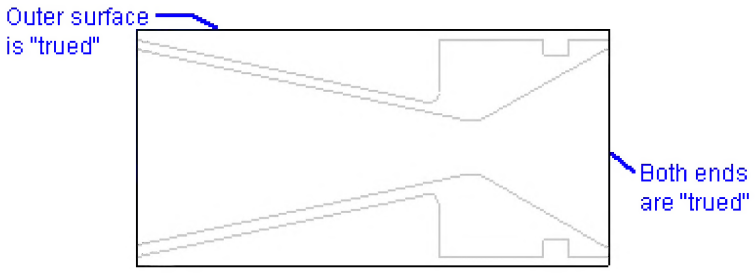



Figure 4 -- Truing the ends and outer surface
Note: Black represents workpiece outline. Grey represents finished nozzle outline

4. Next, that portion of the nozzle that interfaces with the motor casing is **turned** to blueprint size, as shown in Figure 5. Before making any cuts, measure the existing diameter (D) of the workpiece. The amount (thickness) to be turned off (t) is always given by

   **t = (D - Df)/2** where Df is the final (blueprint) diameter

   In other words, *the diameter of the workpiece is reduced by twice the amount being cut off.* Perform the turning operation, removing a suitable amount of material each pass. For my lathe, I typically remove 10-15 thou during each the roughing cuts. If, for example, the calculation indicated 55 thou needed to be removed (t = 0.055"), this would indicated 5 cuts of 10 thou, followed by a single cut of 5 thou. However, the actual cut may differ *slightly* from 10 thou, so stop the lathe when the final diameter is nearly reached, take a diameter measurement, and recalculate the amount remaining to be turned off. Then make the final cuts based on this, removing no more than 5 thou per cut.

   * A "thou" is machinist talk that refers to 1/1000th of an inch, or 0.001 inch. In metric, this is approximately equal to 1/40th of a mm, or 0.025 mm.

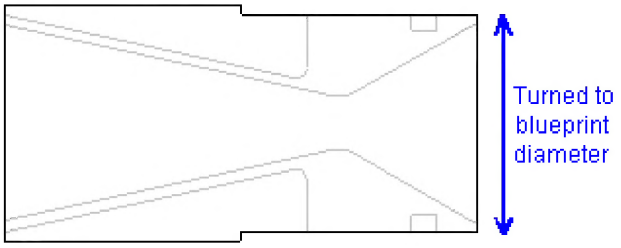



Figure 5 -- Turning that portion of the nozzle that fits into the motor casing
Note: Black represents workpiece outline. Grey represents finished nozzle outline

---

**Example**: After truing, diameter of workpiece is D = 1.240". Blueprint diameter is 1.080". The amount to be turned off is (1.240-1.080)/2 = 0.080, or 80 thou. If each pass cuts a nominal 10 thou, 8 passes will be required to be performed. After doing 7 passes, take a diameter measurement to find the exact amount to be further turned off. If the diameter measurement is then 1.102, the remaining amount to be turned off is t = (1.102 - 1.080)/2 = 0.011 or 11 thou. I'd then make a cut of 8 thou, re-measure, then make the final cut.

---

5. The o-ring groove(s) is machined next. Make sure that the workpiece is mounted with minimum cantilevered length (sticking out of the chuck) yet allowing sufficient clearance between the parting tool and the chuck. This is to ensure that the workpiece is as rigidly supported as possible, as the groove cutting operation is tricky and vibration (chatter) may occur. A commercial parting bit is typically 0.090" wide, so after cutting the initial groove, the tool must be moved and a second cut made to widen the groove. Cutting should be done at a rotational speed significantly slower compared to that used for turning. Feed rate should be moderate. If too large a cut is attempted, the tool may bite into the workpiece, jamming, and possibly bending the workpiece (this has happened to me more than once). Make sure that the parting tool is very **sharp**, with the proper cutting angles and clearance angles. The cutting edge should be located **slightly below** the centerline of the workpiece. If the tool is a little too high it will have a tendency to 'climb' the work. Depth of the groove is simplest to determine indirectly, by using the calipers to measure the diameter of the grooved section. Otherwise, use a depth gauge to measure directly. Most calipers have a built-in depth gauge. After the groove has been cut to blueprint depth, chamfer (break) the two sharp edges of the groove (a fine file works well).

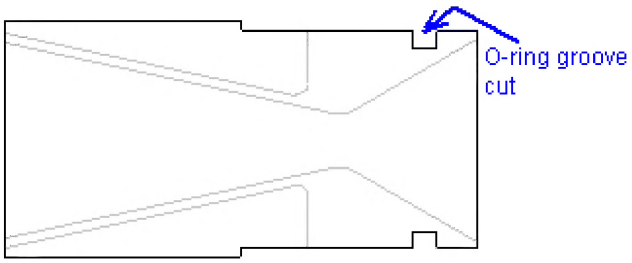



Figure 6 -- Cutting the O-ring groove
Note: Black represents workpiece outline. Grey represents finished nozzle outline

6. Next, using a centre drill bit mounted in the tailstock drill chuck, drill a centre hole into both ends of the workpiece.
7. Drill a **hole through the workpiece** of a diameter of the nozzle throat. It is best to drill approximately half way, then turn the workpiece around, and drill the remaining depth. Use plenty of cutting fluid, and clean out the flutes of the bit regularly. If the throat diameter is quite large, it may be prudent to drill a smaller diameter hole(s) through the workpiece first.

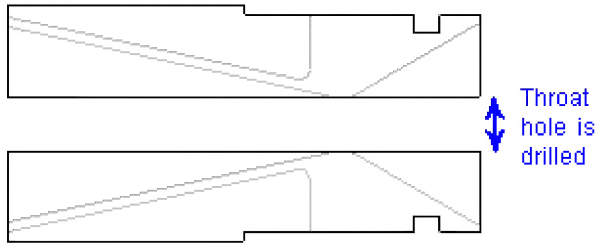



Figure 7 -- Drilling a hole through the workpiece equal to the nozzle throat diameter
Note: Black represents workpiece outline. Grey represents finished nozzle outline

8. The **convergent and divergent interior cones are machined** next, as illustrated in Figure8.

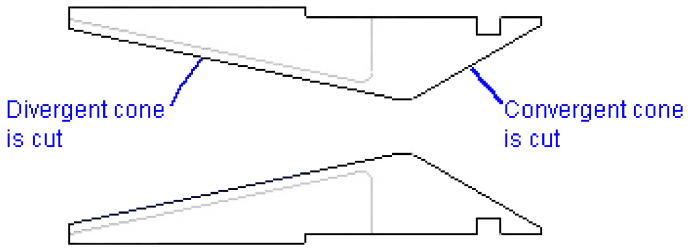



Figure 8 -- The inner profiles are bored out.
Note: Black represents workpiece outline. Grey represents finished nozzle outline

Two approaches are possible. One, to use a boring tool bit to cut out the entire conical portions, or alternatively, use a series of increasingly larger drill bits to create a "stepped" cone, then use a boring tool bit to finish to the desired profile.

Figure 9 -- Drilling and boring operations

The second method is the approach that I now use, and I"ve found that it greatly reduces the effort required to form the interior convergent and divergent profiles. As well, it greatly reduces the tedium and time required to fabricate a nozzle. To use this method to full advantage, it is necessary, of course, to have a set of sufficiently large drill bits. Fortunately, an "off-shore" set of drill bits ranging in size from ½" to 1" (in 1/16" increments) is not expensive -- I bought a set for $30 CAD. The investment pays off in spades. Knowing exactly how deep to drill each incremental size is critical. Drilling too deep can write-off a potential nozzle. As such, I've created an *Excel* spreadsheet (NOZLBORE.XLS) that spells out exactly how deep to drill each bit size. This simplifies the process and greatly reduces the potential for

making costly mistakes. The output of the spreadsheet provides the required drilling depth, incorporating a suitable positive depth allowance to tolerate for inaccuracies in drilling. After drilling all the required holes, a "stepped" profile results. I've used this spreadsheet for several nozzles, with excellent results.

Figure 10 --Stepped profile that results after successive drilling

The final blueprint profile is then cut using a boring bar, after setting the compound-slide to the required angle. The blueprint exit diameter (or entrance diameter) is used as the criterion to determine when sufficient material has been bored out. When boring, the rotational speed should be the same as that for turning, or slightly higher (good to experiment, too high a speed while cause high-pitched squeal or chatter). Cutting depth should be small, only a few thou at a time. Trying to cut too much will cause the inherently flexible boring bar to simply deflect (bend) and not make the full expected cut.

9. Once the interior profiles have been bored out, the worst is over. It's time to celebrate, as it's pretty much clear sailing from here on! The next to last step of the latheing operation is to **turn down the outer profile** of the divergent cone, as shown in Figure 11.

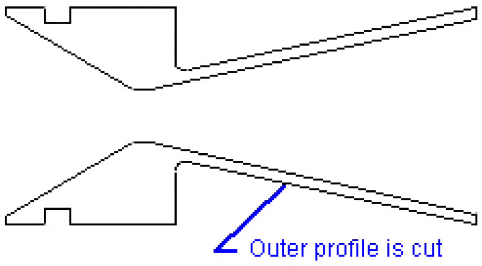



Figure 11 -- Turning the outside profile of the divergent cone
Note: Black represents workpiece outline. Grey represents finished nozzle outline

To perform this step, the compound-slide is first set at the required divergent angle, then the turning operation performed. This is a straightforward operation and can be completed relatively quickly by taking successive cuts parallel to the axis of the workpiece with the aid of power cross-feed.

Once the material becomes quite thin, the workpiece may begin to vibrate at a high frequency and either screech or ring like a bell. This can result in poor cutting and a poor finish. To eliminate this vibration, simply stuff a wad of putty or plasticine into the divergent portion of the nozzle.

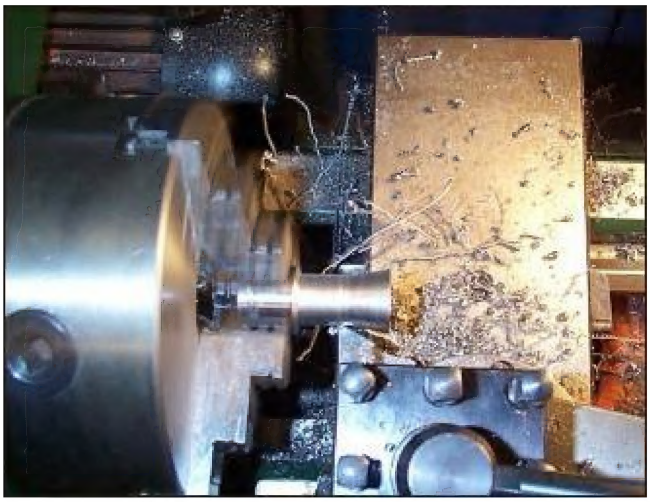


Figure 12 -- Using the power carriage feed to cut metal quickly and easily

10. The final latheing operation involves finishing the nozzle using emery paper to **polish the flow surfaces** inside the nozzle, and to round the throat entrance. This latter step is important, as the efficiency of the nozzle is strongly influenced by how the flow of combustion products occurs in this region. The entrance to the throat is the region of highest acceleration of the products anywhere in the nozzle. It is desirable to minimize the flow acceleration in order to reduce particle velocity lag associated with two-phase flow (see the *Theory* webpage on Two-Phase Flow for a more complete discussion on this important topic). The method that I use to round the throat entrance is to wrap emery cloth around a wooden dowel (of diameter slightly less than the throat). Set the lathe to turn at its maximum rotational speed. At this time, the outside surfaces of the nozzle may be polished to remove the slight roughness inherent with the turning operation.

11. The only task remaining before the nozzle is completed is to **drill and tap the attachment screw holes** (unless, of course, snap rings are being used to retain the nozzle). Since it is important that these holes line up precisely with the corresponding attachment holes in the motor casing, it is best to temporarily assemble the two before drilling. Installing an o-ring helps to hold the nozzle firmly in place, but use an old one, as it will likely get damaged (nicked) during disassembly. Lubricate with grease to ease installment of the o-ring. The next step is to accurately mark the locations of the holes on the casing (remember - the holes should always be at a distance of at least 1.5xD from the edge, preferably 2xD, where "D" is the hole diameter). This can be accomplished by cutting a strip of paper (say, 1 cm. wide) and wrapping it once around the casing. Using a pencil, place a mark indicating where the strip crosses over the starting end of the strip. This identifies the length of the casing **circumference**. The strip of paper can now be place flat and, using a ruler, divide this circumference length into the same number of equal parts to that of the number of holes required. For example, if 6 holes are needed, divide the length by 6. Place pencil marks on the strip indicating each of the increments, then re-wrap the paper around the casing and transfer these marks to the casing.

---

**Example**: A total of six 5 mm holes are required for attachment of a nozzle. Using the technique described above, the circumference of the casing was found to be 120 mm. Dividing the circumference by 6 results in 6 equal increments of 20 mm. Place additional marks on the strip at 20, 40, 60, 80, 100 mm locations. Using an edge distance of 2xD, the holes are to be drilled 2x5=10 mm from the end of the casing.

---

Next, centre punch the locations. Drill a hole at each location into the casing and nozzle, starting off the drilling process with a centre drill bit. After this pilot hole has been drilled (it need only be shallow), disassemble the nozzle from the casing. The holes in the nozzle are next drilled to blueprint depth of the correct diameter for the thread size. If blind holes are

being threaded, then tap the holes using a *bottoming tap* in order to thread the holes more completely.

Figure 13 -- Tapping the nozzle attachment holes

Use plenty of lubricant and perform the tapping operation slowly, reversing the direction of the tap very frequently to break the chips. Turn in ¼ turn, then out ¾ turn, repeating until the hole is completely threaded. Small taps can fracture easily, either through inadvertent bending or by applying too much torque. A broken tap is impossible to remove -- it cannot be drilled out, don't even try, as you'll ruin the nozzle in the effort. If the tap should happen to break, grind away the broken stem until it is flush. Drill new holes and try again. **Only use good quality HSS taps, never carbon steel** which are particularly prone to breakage.

## Machining Graphite

Due to its high temperature capabilities, graphite is an excellent material for rocket nozzles, especially for hot burning propellants. Two drawbacks are graphite's relatively low strength and brittle nature. Graphite stock is also quite a bit more expensive than metal stock. For these reasons, graphite is usually used to make a throat insert, rather than a complete nozzle. Figure 14 illustrates a graphite throat insert. Inserts are fitted into a metal nozzle shell, usually as a tapered plug fit.

Figure 14 -- Aluminum nozzle with graphite throat insert

My experience with machining graphite is fairly limited, and is on-going. What follows is based on this limited experience. Graphite machines well, and larger cuts can be made compared to metal. However, graphite stock is an abrasive material, and tool bits tend to dull rather quickly. Although graphite is a natural lubricant, apparently it is the binder that gives graphite stock an abrasive nature. Since dull tool bits affect accuracy of a cut and the quality of the finish, bits should be re-sharpened frequently. For this reason, tungsten carbide is a better choice than HSS. In industry, diamond coated tool bits are sometimes used to machine graphite stock to extend tool life and for reduced cutting friction.

The cutting operation produces tiny chips accompanied by a very fine dust. The dust is quite pervasive, and besides making a mess, can damage electric motors and switches due to its electrical conductivity. As such, it is paramount to control the dust. The method I use is to vacuum up the dust as it comes off the cutting bit, using a wet/dry shop vac. This particular vacuum cleaner has a high-capacity, multi-pleated filter that can tolerate a large intake of fine dust without impeding air flow. As well, the removable pleated filter can be readily cleaned. An even better solution to dust capture may be a disposable dust bag that fits over the pleated filter. Such bags are available for many shop vacs.

I devised a simple solution to avoid having to manipulate an unwieldy vacuum hose and crevice tool around the cutting action. A length of 3/8 inch (ID) flexible vinyl tubing is simply inserted into the mouth of the crevice tool (do not further block the intake, as the vacuum motor requires unimpeded air flow for cooling!). The other end of the tubing is then placed near the cutting action for dust pickup. The tubing can be conveniently taped or tie-wrapped to the toolpost, or manipulated by hand. This solution works well to suck up the fine dust. The larger chips fall into the lathe bed for later cleanup.

Graphite chips should be thoroughly cleaned from the lathe components because of the potential for wear of moving parts (e.g. ways, feed-screw) due to the abrasive nature of the chips. Graphite dust that gets on your hands, or on the floor, cleans up well with soap (an effective emulsifier) and warm water.

Graphite stock is usually cut dry. Oil-based lubricant should not be used (messy). I have heard of water being used to keep down the dust, however, I would not recommend this due to the potential for rusting of the lathe parts.

The cutting bit should be ground with a zero rake angle to reduce flaking and consequently improve surface finish quality.

## Safety

Although I've never been injured using a metal lathe (other than the odd nick or burn), I've certainly recognized that a lathe holds many potential hazards, and as such, deserves due respect. The spinning chuck and potential for the workpiece to fly off are two obvious potential hazards. The razor sharp swarf that is produced from the workpiece can, and sometimes does, get caught up in the spinning chuck. Do not wear cotton gloves or loose clothing (especially not a necktie!). A short sleeve shirt is probably better than a long sleeve shirt (beware of hot metal chips, though).

The following is a listing of some of the more basic guidelines regarding the safe use of a metal lathe. Of course, using common sense, being alert, and paying close attention to the job being performed are essential.

- Never leave a chuck key in the chuck for even a moment. If the lathe were started up with the key in the chuck, it will fly off and become a formidable projectile, often targeting the operator's face. Make sure the lathe is switched off before chucking or unchucking the workpiece.

- A metallic item of mass rotating at high speed represents a significant amount of stored energy. A chuck, on its own, is heavy, and when holding a workpiece, even more so. Jamming,

dismounting of the workpiece, or other such adversity may subject the operator to the unleashing of this energy, often manifested in a sinister way. So beware, be alert!

- ALWAYS WEAR EYE PROTECTION WHEN USING A LATHE OR ANY OTHER MACHINE TOOLS. In fact, I wear eye protection whenever I perform any operation that involves an exchange of kinetic energy, for example, using a centre-punch or even using a hacksaw.
- Keep your hands away from the rotating workpiece and cutting tools. It is tempting to pull away the "thready" metal swarf as it forms at the cutting tool, but just imagine what would happen if the swarf suddenly got caught up by the rotating chuck (hint: the swarf is razor sharp).
- Don't reach over or put your hands anywhere near the rotating chuck.
- For filing operations, hold the tang end of the file in your left hand so that your hand and arm are not above the spinning chuck.
- Never use a file with a bare tang - if it gets caught, it becomes an effective hand-impaling weapon.

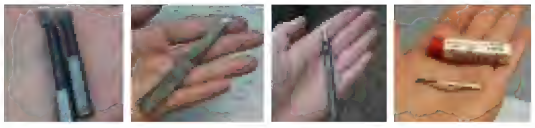
## Appendix A

Custom-made boring tools

**I.** If the throat diameter of the nozzle is quite small, such that a standard boring tool can't quite get into the confined space, it is often possible to modify the tool bit such that it can accomplish the job. It is simply a matter of **sculpting** the tool bit around the cutting edges to remove excess material in order to give the needed clearance. This is done using a power grinder. It is important to prevent the cutting edges of the bit from getting hot. As such, the bit should be regularly dipped into a pan of water. To make certain enough material has been sculpted away, position the tool bit into the nozzle near the throat, and inspect closely for interference. As well as removing material from the sides of the bit, grind the bottom surface, especially at the front end of the tool bit.

**II.** Several years ago, after realizing that the boring tool that I was using to make a nozzle was simply too big to machine the throat region (even when all excess material around the cutting edges was ground away), I searched the workshop for something to make a small boring tool from. I came across an old, small flat file, roughly ½" wide by 1/8" thick. Perfect. I carefully ground away the tip to reduce the width, and improvised cutting edges. I found that this tool worked well, and used it to successfully machine the finishing cuts of the nozzle where the standard boring tool could not reach. The only major drawback with this tool, being of carbon steel rather than HSS, is that it tends to need resharpening often (thereby shortening its useful life).

**III.** More recently, while fabricating a nozzle for the new *A-100M* motor, I decided to try to make a boring tool using a HSS tool blank, which should be capable of keeping a sharp edge for a longer time. I had recently purchased some small 1/8" square blanks, just about the perfect size for boring the ¼" diameter throat. To hold the tool bit, I utilized a length of 3/8" hex steel bar, into which a suitable diameter hole was drilled. Into this hole, the bit could be inserted with a snug fit. To prevent rotation of the bit, I drilled & tapped two holes and installed #6-32 Allan (set) screws. Cutting edges were added and excess material ground away from the tip to maximize clearance when cutting in the confined region at the throat. This boring tool was found to work very effectively.



*Left*: Sculpted boring tool bit, compared to a similar "as purchased" tool bit.
*Middle*: Boring tool made from a flat file
*Right*: Boring tool made from HSS blank, and holder.
*Far right*: Solid carbide micro boring tool, purchased from KBC Tools.

## Appendix B

Reaming

As an alternative to boring, a *tapered reamer* can be used to cut the inner profile of the nozzle divergent cone. A tapered reamer is a cutting tool bit that has tapered cutting edges. Reamers, which are made from HSS, can be quite expensive, especially for the larger sized ones that are suitable for nozzle work. I happened to come across such a reamer during a recent visit to Princess Auto & Machinery, which specializes in surplus tools and machine parts, that was affordably priced ($20 CAD). This particular reamer has a $7.5^{o}$ *half-angle*, which is quite a lot more shallow than the conventional $12^{o}$ *half-angle* that I usually use for my divergent cones. The main drawback to such a shallow angle is the increase in nozzle length, and therefore weight, that results.
I fabricated an adapter to secure the reamer to a standard MT3 (morse taper) fitting that plugs into the lathe tailstock. The completed reamer assembly is shown in the figure below.

I have experimented with using this reamer on steel nozzles and on graphite nozzles. For graphite nozzles, it works exceptionally well. It is only necesary to cut a pilot hole through the length of the nozzle, then use the reamer to enlarge the hole and form the tapered divergent cone. This process is illustrated in the photo below.

For a steel nozzle, it is necessary to use the method described earlier of drilling increasingly larger holes to create a "stepped" profile. This serves to remove most of the material. The reamer is then used to finish cutting the divergent cone. I have found that the reamer works very well, with essentially no chatter or other difficulties. Plenty of coolant is used to help maintain the tool's sharp

cutting edges.
Now, if I could only find an inexpensive 12$^{o}$ reamer...!

## Appendix C

Useful Links

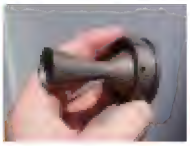
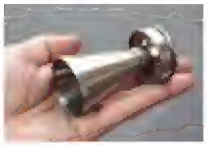
**Virtual Machine Shop** - A must-see site, includes video clips explaining parts of the lathe and various machining operations : Engine Lathe - Topics

Other links :
Introduction and glossary
Lathe construction
Turning operations
Facing operations
Parting operations
Drilling operations

Turning speeds & feeds
US Army Training circular TC 9-524 "FUNDAMENTALS OF MACHINE TOOLS"
Grinding lathe tools
Grinding your own lathe tools
Cutting & boring tools (PDF file)

## Appendix D

Some of the rocket nozzles I have machined

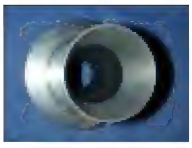



*A-100M* *Epoch* *Juno* *Lambda* Graphite

Steel nozzle shell with graphite insert

---

Hendrik Lau's *HL6000M* aluminum alloy nozzle with graphite insert



---

***Sugar Shot to Space*** project:

## Appendix E
## Tapping Misadventures

Tapping (threading) holes for installation of machine screws is a common and simple operation. Nevertheless, a tapping bit will occasionally break during tapping, despite care being taken. Not only is a broken tap an aggravating occurance, it can be quite problematic. This is especially true if the workpiece represents a large investment in time and effort. An excellent example is a rocket nozzle. I have experienced this mishap on occasion despite my awareness and best efforts at careful tapping. The smaller the tap size the more vulnerable it is to breakage, especially when tapping a harder material such as steel. Taps larger than #10 screw size are not readily broken. I have learned to be particularly careful when tapping small holes by minimizing applied torque and by avoiding any bending that could overstress a tapping bit. Nevertheless, I have experienced breakage even when administering such care. My own theory is that the breakage is often a result of metal fatigue, that which is known as "low-cycle fatigue". A product of unavoidably high stress, stress concentration and tiny intrinsic flaws in the bit, low-cycle fatigue can lead to microscopic crack growth followed by sudden breakage even while being lightly loaded. I have noticed a correlation between such breakage and amount of usage a tapping bit has undergone.

### Avoiding Breakage

Harm caused by a broken tap can generally be repaired. However, it's often not easy, and it is best to avoid such breakage altogether. One means of avoidance is to use a larger screw size and reduce the potential for breakage. However, with rocket nozzles, this is not an option in many cases. Another means of avoidance is to "life-limit" a tap. In other words, for critical applications a particular tap would only be used a limited number of times (10, for example), then placed into "quarantine". These taps could still be used, but exclusively for non-critical applications. Another means of avoidance is to employ softer metals, such as aluminum alloys, rather than steel, whenever possible. Generous use of suitable purpose-made lubricant (cutting oil) is also important. Taking small cuts and "backing out", as well as removing the tapping bit and cleaning off the bits of swarf -- an especially important practice when tapping blind holes.

### Repairs

If a tap breaks and enough shank is protruding from the hole, it can sometimes be removed by gripping the shank with a pair of vise-grips or a bench vise, and backing out the tap. Because a tap is made from very hard, brittle material, this does not always work well. What often happens is that the tap simply breaks (shatters) while being gripped.

If insufficient stem is protruding from the tapped hole, the tap can occasionally be removed by using a small punch or the corner of a cold chisel and carefully tapping on one side of the tap in such a manner as to "unscrew" it.

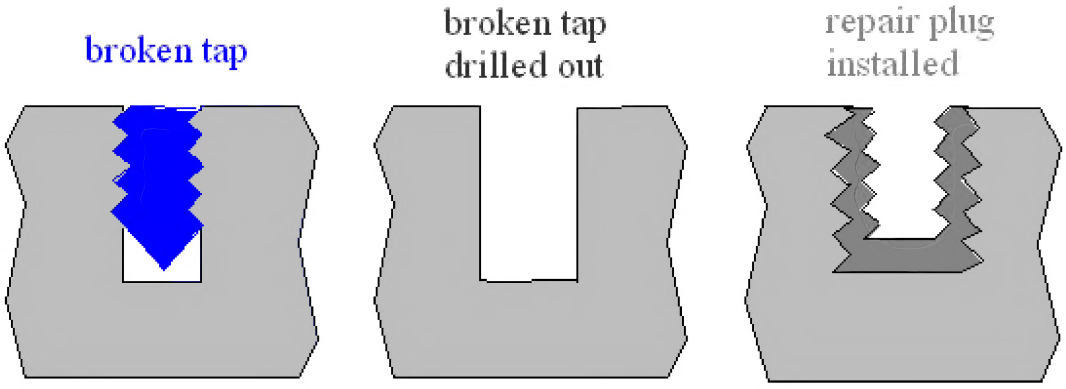
If these means fail, a solution to the problem may be to simply leave the broken tap in place and grind the protruding shank flat with a grinding stone and Dremel tool. This would necessitate drilling or tapping a replacement hole in an alternate location. This solution is, needless to say, often not a viable one.

Sometimes it is imperative that the hole be salvaged and thus the broken tap be removed. Tapping bit material is extremely hard, especially if it is high-speed steel, and a normal drill bit will be ineffective. However, a carbide-tipped drill bit will readily drill out a broken tap. Purpose made drill bits are available but are rather expensive. I have used a common masonry drill bit as an inexpensive alternative. The carbide cutting tip should be sharpened first using a green silicon carbide grindstone (masonry drill bits are usually rather dull). Plenty of lubricant should be used when drilling out a broken tap.

The consequence of drilling out a broken tap is a larger hole size, meaning that the hole will need to be opened up and threaded for a screw that is one size larger than originally planned. For example, if the original screw size is #6, then a #8 screw would be installed. If this is not an option, then a repair plug must be installed.

**Repair Plug**

After drilling out the broken tap, the hole is opened up for installation of a plug. A plug is simply a short length of externally threaded steel (or aluminum). This can be a length of threaded rod or the threaded portion of a machine screw. The diameter of this plug should be large enough to accommodate a drilled and tapped hole for the originally chosen machine screw. The discrepant hole is drilled to the required size to tap the hole to accept the plug. After cleaning out the hole to remove all traces of swarf and oil, epoxy (such as J-B Weld) is applied to the threads of the plug and it is screwed firmly into place in the repaired hole. After the epoxy has cured, the plug is cut and ground flush with the surface of the workpiece. The plug can then be drilled and carefully tapped to accommodate the originally planned machine screw. This process is illustrated in the figure below.




## Appendix F
## Snap Ring Grooves

Snap Rings, also known as Retaining Rings, are an excellent alternative to machine screws for retaining a nozzle or bulkhead. An internal snap ring is installed into a groove machined into the motor casing. The concept is illustrated in Figure F1.

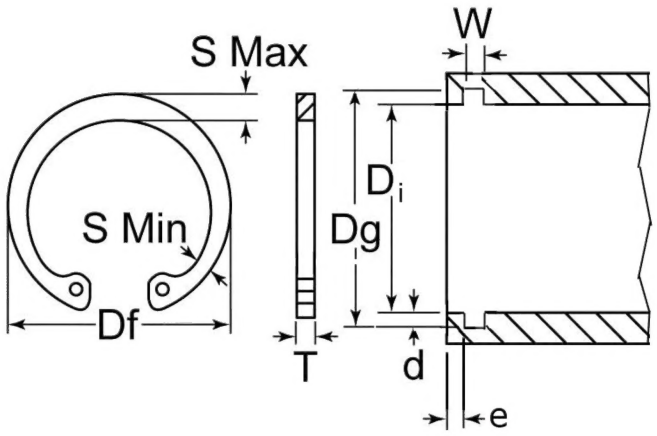



Figure F1 -- Snap Ring installation

The snap ring groove, of width "W" is machined into the wall of the motor casing to a depth "d". To cut a Snap Ring groove the casing is mounted in the lathe chuck as deep as the chuck will allow. For longer casings it may be necessary to use a steady rest to support the free end of the casing. A Snap Ring groove cutting bit is then used to cut the groove to the required depth. Figure F2 illustrates the type of cutting tool that I use. It is made of solid carbide. Although fairly expensive ($35 USD), it lasts "forever" and does the job admirably.

Figure F2 -- Snap Ring groove cutting bit

For metallic motor casings, the maximum groove depth is 1/2x the wall thickness. For my motors that use a thin-walled casing of 0.065 inch (1.65mm) I cut the groove 0.030 inches deep (0.76mm). The width of the groove is not critical. I typically make the groove width to be 1-1/2 times the thickness, "T", of the Snap Ring.

The width of the lip, dimension "e", is important from a structural perspective. The width of the lip must be sufficient to prevent the lip from tending to shear off and tending to swell ("flare") due to stress arising from axial load as a result of motor chamber pressure. The minimum lip width (inches or mm) is given by:

$$e_{min} = \frac{P\ D_i}{F_{su}}$$

where P=maximum expected chamber pressure (psi or MPa), Di=casing inner diameter (in. or mm) and Fsu=shear strength of casing material (psi or MPa). If the motor casing is 6061 aluminum alloy, use Fsu=26000 psi. This method provides for a conservatively sized lip with a generous safety factor. *Example*

---

**Cutting snap ring grooves** in *Impulser* motor casing
Image 1 Image 2 Image 3

**Originally posted June 27, 2004**
**Last updated July 11, 2018**

Return to Top of Page
Return to Index Page